Title	NUMERICAL SIMULATION OF FRACTURE MECHANICS USING THREE-DIMENTIONAL FINITE ELEMENTS
Author(s)	Ruggieri, Claudio
Citation	大阪大学大型計算機センターニュース. 84 p.57-p.59
Issue Date	1992-02
oaire:version	VoR
URL	https://hdl.handle.net/11094/65955
rights	
Note	

NUMERICAL SIMULATION OF FRACTURE MECHANICS USING THREE-DIMENSIONAL FINITE ELEMENTS

Claudio Ruggieri [1]

Structural analysis based on a fracture mechanics approach has recently emerged as a valuable method for engineering design and structural integrity assessment, where a crack or flaw is introduced into the structure in order to calculate the local stresses and strains leading to failure. Although well-developed theories can describe the mechanical behaviour of linear materials in simple geometric and loading configurations, most of the engineering structures present a high degree of nonlinearity and complex geometries, which can only be solved by finite elements[1].

Recent developments in general purpose large-scale nonlinear finite element programs have provided signicant capability for analysing full three-dimensional models. The physical phenomena involved, however, can be very difficult to simulate and yet be very demanding of computer resources. The use of very fine meshes in the crack region are often necessary and large numbers of load increments may be required to solve the problem. This can make the computational analysis so large that only supercomputers provide practical solutions and fast simulations to be used in engineering applications.

Finite Element Analysis in Nonlinear Problems

The use of finite element methods in mechanical and structural design generally follows the process presented in figure 1[2] . The mechanical idealization involves static and kinematic assumptions for analysis purposes that lead to differential equations governing the structural model. The finite element solution of the model depends largely on the numerical procedure employed, mainly on highly vectorized iterative solution strategies which have dramatically increased efficiency in three-dimensional implicit calculations[3].

The three-dimensional modelling that is currently in progress at the Laboratory of Mechanics on Materials and Structures uses the implicit code NIKE3D[4]. A somewhat modified version (JNIKE3D) has been implemented on the NEC supercomputer SX-2N at the Computation Center, Osaka University. The numerical procedure employed is fully nonlinear and large rotations and strains can be accomodated. A library of constitutive algorithms incorporating a wide variety of nonlinear material models is also available, but only 8-node solid, 4-node shell and 2-nodes beam elements are used.

[1] Laboratory of Mechanics on Materials and Structures
Department of Welding and Production Engineering, Osaka University
Phone : 06-8775111 Ext 4818
E-mail : a62420a@ccsun01.center.osaka-u.jp

The incremental-iterative numerical algorithm implemented in NIKE3D follows a stress rate formulation. The solution at load increment $n+1$, given the solution at load increment n, is obtained by imposing the equilibrium of internal nodal forces with the external applied load. Generally, the numerical procedure is based on quasi-Newton methods and a very large system of linear equations has to be solved at each load increment, which involves heavy computational burden in both storage and CPU.

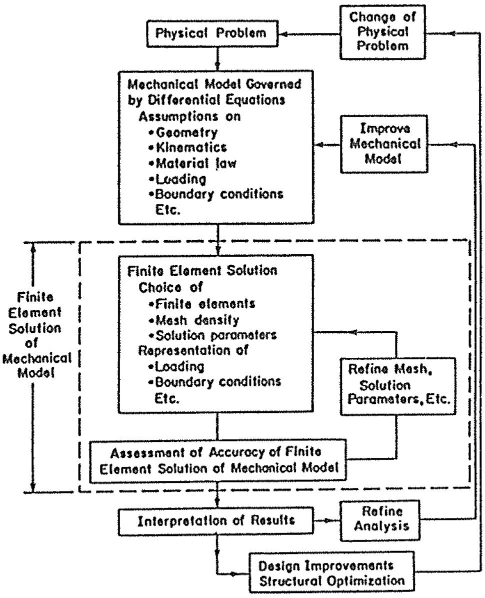

Figure 1 - The process of finite element analysis.

Computation of the Crack Model

The three-dimensional mesh of the crack model was built with the aid of a mesh generator available at the laboratory's work station . Because of symmetry, only one quarter of the model was analysed. The mesh consisted of 876 elements, 1190 nodes and 3234 degrees of freedom (DOF). The model was a high strenth steel plate of 150 mm length, 30 mm wide and 15 mm deep with a sharp edge crack of 3 mm length. The minimum element size was 0.1x0.1x0.25 mm. Figure 2a presents the three-dimensional mesh of the crack region.

The static analysis was performed by using an elastic-plastic material model and imposing a total displacement of 7.5 mm at the loading point. In order to achieve numerical convergence, the displacement increments were varied according to the deformation level, so that 1200 increments were required to fully solve the specified deformations. The total solution time (CPU) took 140 minutes on the NEC supercomputer SX-2N, which most of them were spent to solve systems of linear equations directly by Gaussian elimination. Figure 2b presents the deformed shape of the crack region at the maximum applied displacement.

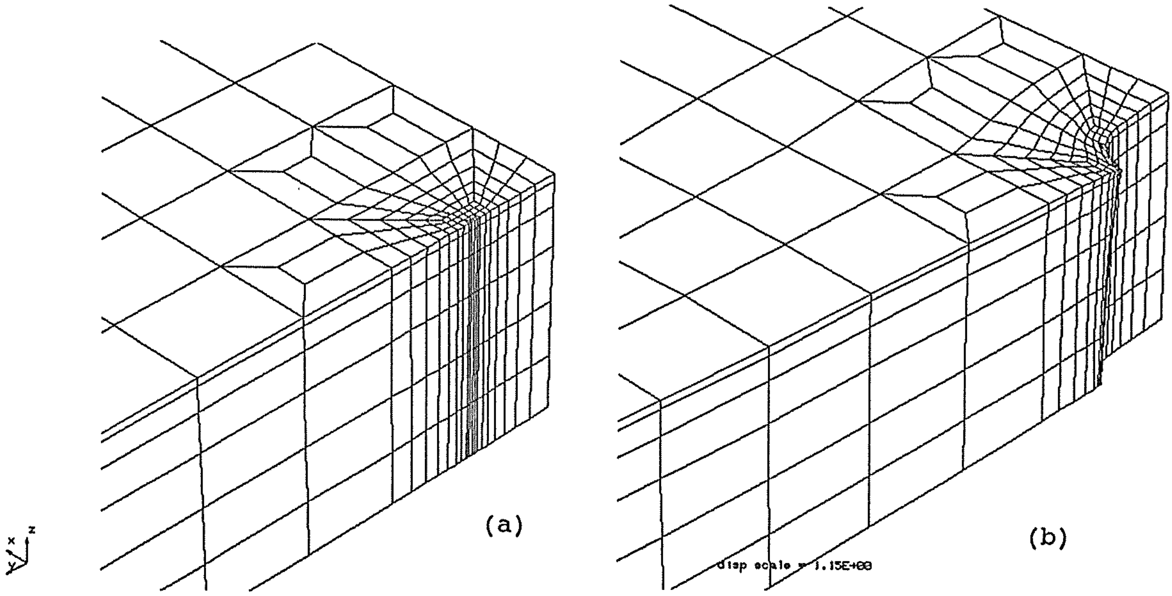

Figure 2 - Three-dimensional mesh (a) and deformed shape of the crack region at the maximum applied displacement (b).

Summary

A numerical simulation of a cracked body was performed by using a general purpose large-scale nonlinear finite element code. Because of the stress and strain singularity, represented by a sharp crack, the problem presents a high degree of nonlinearity which is very demanding of computer resources. However, the use of the supercomputer at the Computation Center, Osaka University, made full three-dimensional modelling not only possible but also provided quick solutions to be used in engineering analysis.

References

1. Zienkiewicz, O.C. The Finite Element Method. McGraw-Hill, 1977.
2. Bathe, K.J. Finite Elements in CAD and ADINA. Nuclear Engineering and Design, 98:57-67, 1986.
3. Hughes, T.J.R. New Directions in Computational Mechanics. Nuclear Engineering and Design, 114:197-210, 1989.
4. Hallquist, J.O. NIKE3D - An Implicit, Finite Deformation, Finite Element Code for Analyzing the Static and Dynamic Response of Three-Dimensional Solids. University of California, UCID-18822, 1984.